Plant Archives Vol. 19, Supplement 2, 2019 pp. 377-383 **e-ISSN:2581-6063 (online), ISSN:0972-5210**

STUDYING THE HYDRAULIC CHARACTERISTICS OF UPVC BUTTERFLY VALVE BY CFD TECHNIQUE

H.M. Imam[1], Sabreen K. Pibars[2], and W.M.M. Soltan[1]
1. Department of Field Irrigation and Drainage Engineering Researches, Agricultural Engineering Research Institute. Cairo, Egypt.
2. Water Relations and Field Irrigation Department, Agricultural and Biological Research Division, National Research Centre, El-Dokki, Cairo, Egypt.
Corresponding author: sabreennrc@yahoo.com

Abstract

Recently, many butterfly valves made of Unplasticized Polyvinyl Chloride (UPVC) were widely produced, which are differs in its dimension, disc design, and material properties than the commonly used metal ones that affect the hydraulic factors especially the head loss coefficient. Two butterfly valve discs different in geometrical design: thick corrugated type I, and streamlined type II were simulated using CFD technique to inspect the effect of valve disc geometric design and opening angle on pressure difference ΔP between the upstream and downstream and head loss coefficient K_θ. Laboratory tests were performed to the pressure difference measured for the two UPVC butterfly valve types at an average 30 m^3/h flow rate at disc opening angle θ 30, 45, 60, 75, and 90°. The water flow field past the two UPVC butterfly disc type designs was simulated using SimScale cloud-based platform. Simulation analysis showed that, ΔP was increased by the decease of opening angle from θ 90 to 30° for both two design types of valve disc. The pressure difference ΔP of disc type II was slightly higher than it is in type I at open angle θ degrees 90, and 75, but by decreasing the disc open angle θ60, 45, and 30 the pressure difference of disc type 2 was lower than it is in disc type I. The thick cross section and corrugated design of the disc type I gave a higher flow turbulence resulting in a noticeable pressure difference values than streamlined design of the disc type II specially by decreasing open angle θ60, 45, and 30. There was a high velocity gradient appeared around the edge of the two discs due to the change of disc curvature. The velocity becomes higher between the discs and pipe surface by decreasing the opening angle due to the smaller of flow area. As a result of the difference in ΔP, K_θ was increased by the decease of opening angle from θ 90 to 30° for both two design types of valve disc. The head loss coefficient $K_{\theta 1}$ values ranged from 0.522, to 108.473, where $K_{\theta 2}$ values ranged from 0.144, to 70.285 at different opening angles. The experimental results showed a relatively similar behavior in pressure difference such as the results obtained from CFD analysis for the two disc design types.

Keywords : Hydraulic Characteristics, Butterfly valve, CFD Technique

Introduction

Recently, many butterfly valves made of Unplasticized Polyvinyl Chloride (UPVC) were widely produced, which is differs in its dimension, disc design, and material properties than the commonly used metal ones. These differences could affect the hydraulic performance factors specially the friction loss coefficient which is important in selecting, sizing of butterfly valves, calculating pump head requirements and evaluating the energy costs in irrigation systems. CFD simulation provides an effective tool to predict the real or near the same magnitude and range of the real-life values of the hydraulic performance of tested components.

Butterfly valve common construction features are the valve body, the disc and shaft, shaft support bushings or bearings, shaft packing, and a means of attaching an operator to the shaft (Lipták, 2006). As the valve's opening angle is increased from 0 degrees (fully closed) to 90 degrees (fully open), fluid is able to more readily flow past the valve (Gite, 2015). In the design of UPVC butterfly valve , the valve body and the disc are made of UPVC and water tightness is assured by a gasket placed in the valve body.

The different formulations obtained by adding suitable additives and stabilizers render the UPVC the most versatile of all thermoplastic materials, allowing it to be adapted to many applications involving fluids under pressure. It is characterized by a combination of lightness, strength, ease of installation, resistance to corrosion, chemical resistance and high stiffness Carleo (2013).

Computational Fluid Dynamics (CFD) methods are concerned with the solution of equations of motion of the fluid as well as with the interaction of the fluid with solid bodies(Blazek, 2001). It continues to be an impressive tool in helping model real world problems. However, CFD can be used to give insight into visualization of complex flows, and fluid flow problems one is attempting to solve. CFD was able to model the overall behavior of fluid flow for an incompressible fluid around a buttery valve at angles ranging from 10- 90 degrees. It is worth noting however, that despite the relative differences that may occur between the simulated and experimental values, CFD simulations can be used to predict values on or near the same order of magnitude and range of the real life values one is seeking. This can be especially useful if experimental models are not available (Del Toro *et al*., 2015).

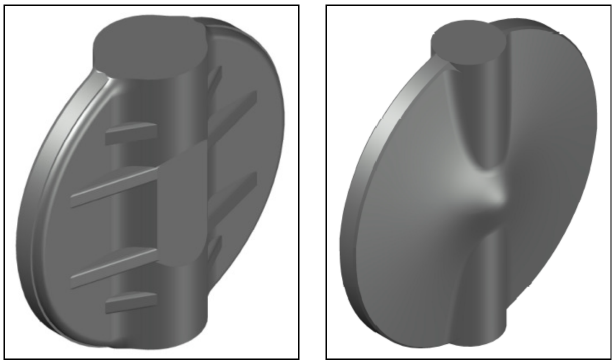
Flow field through butterfly valve is extremely complex and depended on the valve disk geometry, the operating pressure conditions and the disk angle. The flow field may consist of regions of strong pressure gradients, shock waves, flow separation and reattachment (Chaiworapuek, 2007). CFD simulation results show that the disc thickness makes the most significant effect on the flow and structural performance of the butterfly valve. Systematic design with more diverse variables regarding the disc shape in addition to the disc thickness is expected to bring better findings (Kang *et al*., 2014). The loss coefficient directly depends on position of valve disc. As degree of valve disc and discharge increases flow coefficient value goes on increasing. So the flow coefficient loss coefficient are function of valve disc shape (Pisal and Sawant, 2015; and Chaiworapuek, 2007).

The objective of this paper is to determine the pressure drop occurred due water flow past UPVC butterfly disc using CFD technique to have a visual analysis of water flow pattern and to compare the results with the experimental results.

Material and Methods

Experiments

Experiments were carried out at the National Irrigation Laboratory for On-farm Irrigation Devices Testing of Agricultural Engineering Research Institute (AEnRI), Agriculture Research Center (ARC), Dokki, Giza. Two types of UPVC butterfly valve disc of 6” diameter different in geometrical design were tested as shown in fig. (1).



(a) (b)

Fig. 1 : Designs of butterfly valve disc: a. type I, and b. type II.

Both the flow rate and pressure difference were measured using Pressure Drop Test Facility. A general sketch of test facility is shown in fig. (2). The testing bench contains a digital display electromagnetic flow meter with a measurement range from 0 to 100 m^3/h and a relative uncertainty ± 0.2 %. The pressure drop determined through a differential pressure transducer (non calibrated) with a measurement range from 0 to 500 mbar.

Pressure difference was measured in the pipe run 2 times the pipe diameter upstream and 6 times the pipe diameter downstream of the valve. Measurements were taken through two pressure taps located on opposite sides of the pipe at each location.

The average water temperature was 24 °C which make the water density 997.3 kg/m^3 and viscosity 9.1×10^{-4} $N.s/m^2$. The pressure difference measured for the two butterfly valve types at an average 30 m^3/h flow rate depending on available testing facilities of the water source tank and the pump capacity. The pressure difference was measured for a disc open angles 30, 45, 60, 75, and 90 degree. The direct pressure difference measured values are provided in Table (1).

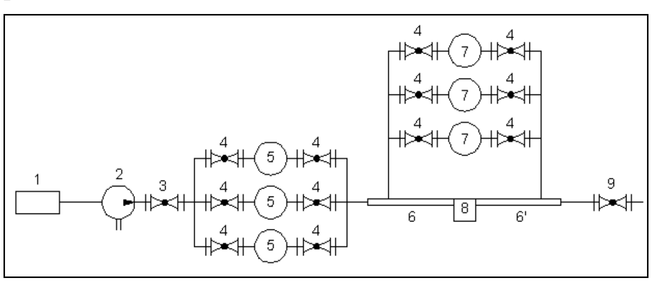


Fig. 2 : General sketch of the principal of the head losses test facility.

1. Water source, 2. Pump, 3. Discharge valve, 4. Manual isolating valves, 5. Electromagnetic flow meters, 6, 6' Set of straight pipes, 7. Differential pressure gauges, 8. Device to be tested and 9. General ball valve.

Fig. 3 : Arrangement of taps pressure upstream and downstream the UPVC butterfly valve.

Loss coefficient:

The flow-resistance coefficient, commonly known as the loss coefficient, *K*, is a dimensionless value commonly used in the design of fluid systems to predict head losses resulting from the presence of various components. The loss coefficient is given as (Del Toro *et al.*, 2015):

$$\mathbf{K_{\theta} = \frac{2gh_{L\theta}}{V_{avg}^2}} \quad (1)$$

where g is the gravity constant of 9.81m/s^2, h_L is the head loss between any two reference points in a system, and V_{avg} = Q/A is the average velocity of the fluid flow, where *Q* is the flow rate and A is the cross-sectional area of flow. The relationship between pressure and head loss expressed as:

$$\mathbf{\Delta P = \rho g h_{L\theta}} \quad (2)$$

The head loss, h_L, is further defined as:

$$\mathbf{h_{L\theta} = \frac{\Delta P_{\theta}}{\rho g}} \quad (3)$$

where ΔP is the pressure loss measured between the upstream and downstream the butterfly valve, and ρ is the density of water. Combining the previous two equations and simplifying gives:

$$\mathbf{K_{\theta} = \frac{2\Delta P}{\rho V_{avg}^2}} \quad (4)$$

Simulation of flow water pattern past butterfly disc using CFD:

SimScale cloud-based platform was used to simulate water flow field past the two UPVC butterfly disc designs. SimScale is a fully cloud-based engineering simulation platform reinvented for the web that allows more efficient, and better results at lower cost. It is a computer-aided engineering product developed by SimScale GmbH. SimScale platform is accessible completely via a standard web browser with an easy to use interface which supports simulation of Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA), thermodynamics. It provides the options to test multiple design versions in parallel and easily share and collaborate projects in real time. The platform supports a large community of engineers, scientists, designers and thousands of companies worldwide (SimScale, 2018).

3D CAD model perpetration:

However, a 3D model of water flow was created for 6 inches size of the three butterfly discs design using AutoCAD 2017 software and exported to ACIS (*sat) file format. It has been assumed a pipe length to be 2 times of the valve diameter for upstream flow and 6 times of the valve diameter for downstream flow for the CAD models. The models created for discs angles 30, 45, 60, 75, and 90 degrees. The model file uploaded to personal account on SimScale platform web site (https://www.SimScale.com/) then a new project was created for every 3D flow model.

Analysis type

The first step for running simulation is to define its general type in terms of its basic physics. The incompressible fluid flow analysis was used to run simulation where fluid density variations are negligible due to the small gradients of velocities and temperature.

Mesh Generation

Mesh is a network that is formed of cells and points. It can have almost any shape in any size and is used to solve Partial Differential Equations. Each cell of the mesh represents an individual solution of the equation which, when combined for the whole network, results in a solution for the entire mesh.

Meshing of the CAD models was generated with Hex-Dominat meshing algorithm, internal meshing model, automatic mesh sizing, moderate fineness, and eight cores possessors.

Material assignment

Material can bee added to the model by clicking on the Add material button. Afterwards specific material parameters have to be defined depending on the chosen analysis type. SimScale platform facilitate a built-in material library with some standard material's properties.

Water was selected as a material then the fluid properties (Kinematic viscosity and density) added automatically to the model.

Boundary conditions assignment

Boundary conditions define how a system (fluid) interacts with its environment. Velocity inlet and pressure outlet were the assigned boundary conditions for the studied model.

Velocity inlet

The velocity inlet boundary conditions define a flow condition in to the domain. The inlet pressure and total (stagnation) values are not fixed but calculated. The inlet velocity set to be 0.4676 m/s according to experiments conditions.

Outlet pressure

Pressure conditions are typically assigned at the opposite end of the model to a flow rate or a different pressure. The pressure outlet boundary condition defines an outflow condition based on the flow pressure (P) at the outlet. This is usually used when there is a flow rate (or velocity), or a higher pressure assigned at the inlet. The assigned value of outlet pressure was 101325 Pa (the value of atmospheric pressure).

Simulation control

Under the tree item Simulation control the different global properties and parameters regarding the simulation process can be adjusted. Here it can also decide on which instance type the simulation is calculated via the number of computing cores setting.

The values of simulation control parameters was set as follows:

Number of computing cores	: 8,
Maximum runtime	: 3000 sec.,
Start time	: 0 sec., and
End time	: 1000 sec.

Simulation runs:

Simulation start running by clicking on the '+' button next to Simulation runs in the simulation tree.

Post-process results:

Once the simulation run is finished, the post processor can be open by clicking on the Post-process results. Cutting plane was added to visualize both velocity and pressure degradation inside the flow domain.

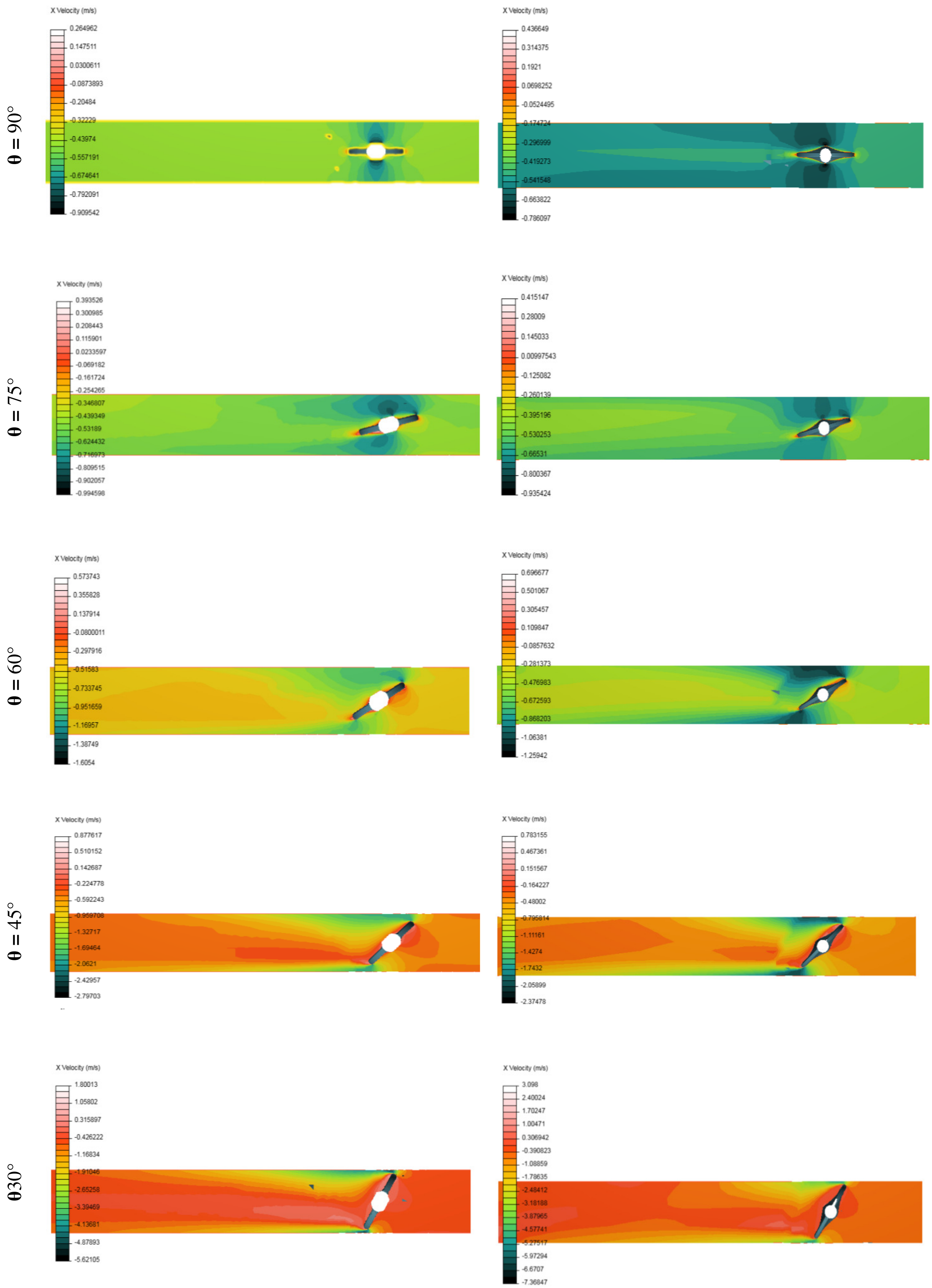
Results and Discussion

CFD analysis of water flow past UPVC butterfly disc:

Velocity Distribution

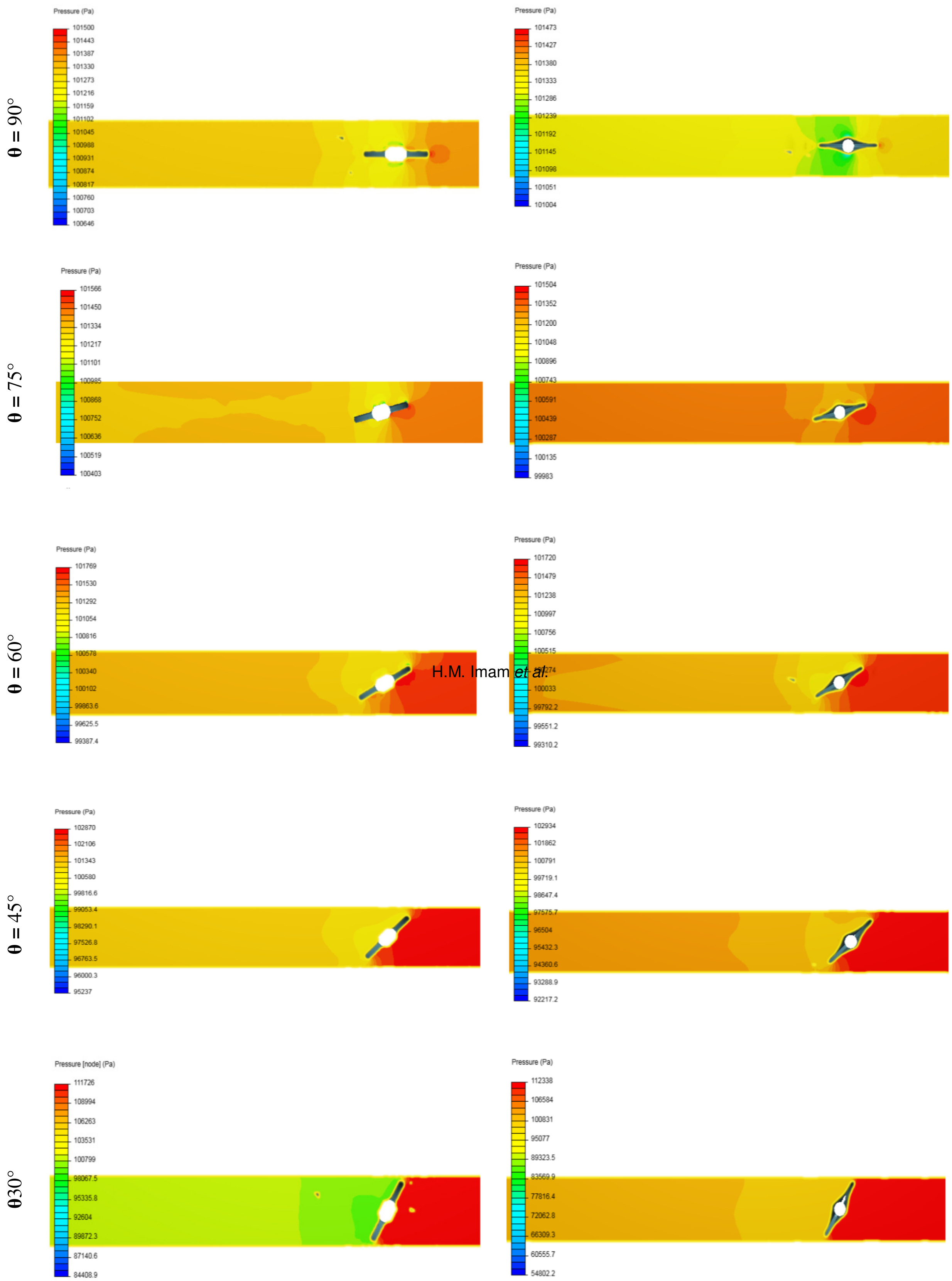
Figure (4) show water velocity distribution past the two types of UPVC butterfly discs. It was found that a high velocity gradient appears around the edge of the two discs due to the change of disc curvature. The velocity becomes higher between the discs and pipe surface by decreasing the opening angle. The velocity will be increased by smaller flow area as shown below.

A small flow separation area, subsequently a smaller vortex formed at rear edge of the disc at θ (90,and 75°). A flow separation and vortex area formed at downstream surface stating from **θ** (60°) and getting wider by decreasing the opening angle θ (45, and 30°).



type I type II

Fig. 4 : Velocity distribution of water flow past the two design types of UPVC butterfly discs at different opening angels θ.



H.M. Imam *et al.*

type I type II

Fig. 5 : Pressure distribution of water flow field around the two design types of UPVC butterfly discs at different opening angels (θ).

Pressure difference

Observing CFD analysis in Fig. (5) of water flow past the two types of butterfly discs it was found that, the pressure difference ΔP between the upstream and the downstream increases by the decrease of opening angle θ. The lowest pressure difference was at θ 90° where the highest ΔP was recorded at θ 30°. Comparing the obtained values of ΔP in fig (5) and table (1) it can be noted that, ΔP values of the valve disc type I was higher than ΔP for type II disc design at θ 90, and 75° then ΔP became relatively closer at θ 60, and 45°. the disc design type I gave a higher value of ΔP than type II at θ 60°.

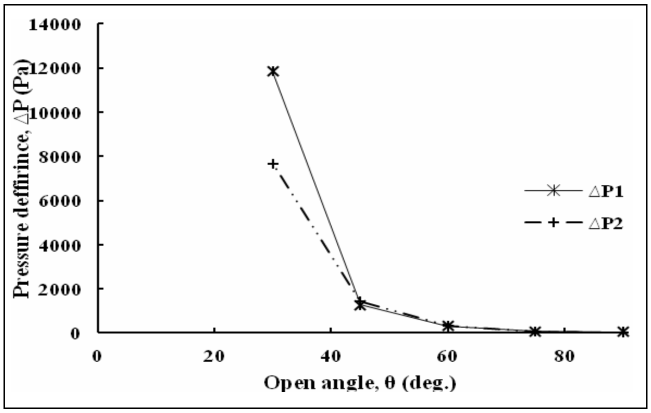


Fig. 6 : The pressure difference (ΔP) of both two disc design types at different opening angles (θ).

Loss coefficient

The resulted pressure difference form the CFD analysis using Simscal platform were investigated to calculate the loss coefficient K_θ of the two UPVC disc types using equation (4). Table (1) show values of K_θ according the pressure difference at every disc opening angle. It was found that, K_θ affected by both the opening angle θ and disc geometric design. The loss coefficient K_θ increases by the decrease of disc opening angle θ. It can be also noted that, $K_{\theta 1}$ of the disc deign type I was higher than $K_{\theta 2}$ of the disc design type II as shown fig. (7).

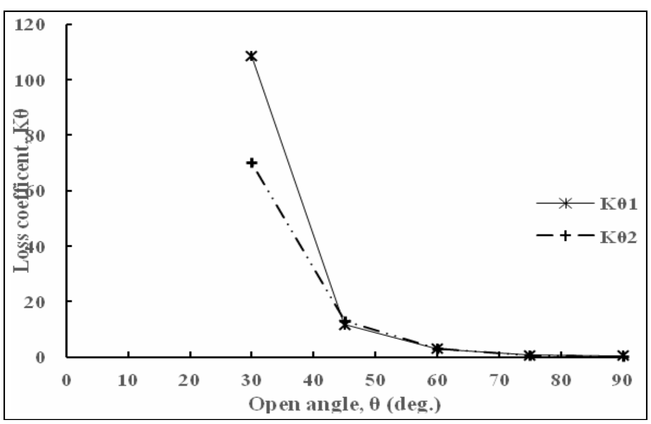


Fig. 7 : The loss coefficient (K_θ) of both two disc design types at different opening angles (θ).

The experimental results

The experimental results showed a relatively similar behavior in pressure difference such as the results obtained from CFD analysis for the two disc design types. The pressure difference between the upstream and downstream the two disc types ΔP increase by the decrease in disc open angle for both the two disc types at the same discharge. The pressure difference ΔP of disc type II was slightly higher than it is in type I at open angle θ degrees 90, and 75, but by decreasing the disc open angle θ60, 45, and 30 the pressure difference of disc type 2 was lower than it is in disc type I as shown in both table (1) and fig, (8). The thick cross section and corrugated design of the disc type I gave a higher flow turbulence resulting in a noticeable pressure difference values than streamlined design of the disc type II specially by decreasing open angle θ60, 45, and 30.

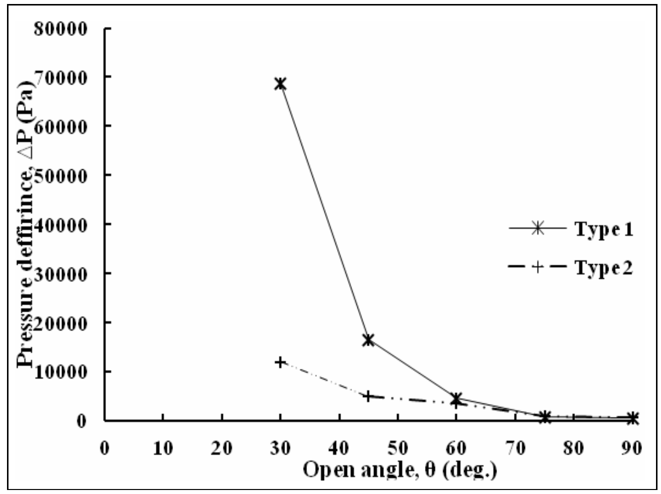


Fig. 8 : The experimental results of pressure difference (ΔP) of both two disc design types at different opening angles (θ).

Simulation analysis showed that, ΔP and as a result K_θ were affected by both the geometrical design and opening angle of the valve disc which agreed with (Pisal and Sawant, 2015 and Chaiworapuek, 2007). It was found that ΔP increased by the decease of opening angle from θ 90 to 30° for both two design types of valve disc. The geometrical design of disc type II gave a better hydraulic performance than the geometrical disc type I (Kang et al., 2014). Observing the CFD velocity analysis in fig. (4), the streamlined of the cross section design gave a smooth and converging increase of water flow velocity past the disc type II with a very small flow separation area at rear edge of the disc at θ90,and 75°resulting a lower ΔP and in turn the loss coefficient $K_{\theta 2}$0.144, and 0.464. A flow separation area formed at front edge by θ60° that increased ΔP resulting in$K_{\theta 2}$ 2.944. By decreasing θ to 45, and 30 the areas of separation become more extensive and the loss coefficient increases (Douglas *et al.*, 2005) $K_{\theta 2}$13.094 and 70.285. The thick cross section and corrugated design of disc type I caused many and wider flow separation areas resulting a higher values of ΔP. It can be noted that the sudden change in disc cross section caused a flow separation area at θ90, 75, and 60°resulting a higher ΔP and the loss coefficient $K_{\theta 1}$ was0.522, 0.709, and 2.913. By narrowing the flow area the velocity gradient had got higher and the separation area became more extensive which increased ΔPat θ45, and 30° and $K_{\theta 1}$ was11.671, 108.473.

Finally, it is worth noting, however, that despite the relative differences that may occur between the simulated and experimental values, CFD simulations can be used to predict values on or near the same order of magnitude and range of the real-life values one is seeking. This can be especially useful if laboratory testing is not feasible (Del Toro *et al.*, 2015).

Table 1 : The experimental results and CFD simulation values of pressure difference (ΔP) of both two disc design types at different opening angles (θ).

θ	Measured values		CFD Modeling			
	ΔP_1 (Pa)	ΔP_2 (Pa)	ΔP_1 (Pa)	ΔP_2 (Pa)	$K_{\theta 1}$	$K_{\theta 2}$
90	498	582	57	15.667	0.522	0.144
75	686	790	77.33	50.667	0.709	0.464
60	4554	3546	317.87	321.253	2.913	2.944
45	16444	4864	1273.588	1428.976	11.671	13.094
30	68656	11942	11837.5	7670.082	108.473	70.285

Conclusion

Recently, many butterfly valves made of Unplasticized Polyvinyl Chloride (UPVC) were widely produced, which is differs in its dimension, disc design, and material properties than the commonly used metal ones. These differences affect the hydraulic factors of the valve specially the head loss coefficient, which is important for irrigation systems designers in calculating the energy requirements and selecting the pump. The effect of both valve disc geometric design and opening angle were investigated on the hydraulic performance of the valve. Two butterfly valve different in disc design type: thick corrugated type I, and streamlined type II were laboratory tested and simulated using CFD technique. The pressure difference measured for the two UPVC butterfly valve types at an average 30 m^3/h flow rate at disc opening angle θ 30, 45, 60, 75, and 90°. SimScale cloud-based platform was used to simulate water flow field past the two UPVC butterfly disc type designs. The visualized results of velocity and pressure distribution of CFD simulation were investigated. Simulation analysis showed that, ΔP and as a result K_θ were affected by both the geometrical design and opening angle of the valve disc. The geometrical design of disc type II gave a better hydraulic performance than the geometrical disc type I. The streamlined disc type II gave a smooth and converging increase of water flow velocity what generates a very small of flow separation area at different opening angle than the thick corrugated type I disc. The pressure difference ΔP between the upstream and down stream of disc design type I was higher than ΔP of disc design type II. As a result of the difference in ΔP, the head loss coefficient $K_{\theta 1}$ values ranged from 0.522, to 108.473, where $K_{\theta 2}$ values ranged from 0.144, to 70.285 at different opening angles θ. The experimental results showed a relatively similar behavior in pressure difference such as the results obtained from CFD analysis for the two disc design types. CFD analysis is a very effective tool in inspecting the hydraulic performance factors of the irrigation system components, that it gives results close to real-life especially with unavailability of suitable laboratory test facilities.

References

Blazek J. (2001). Computational Fluid Dynamics: Principles and Applications, ELSEVIER New York, 3.

Carleo J. (2013). Handbook of PVC Pipe Design and Construction, Industrial Press, INC, 989 Avenue Americas, New York, USA, 7 - 11.

Chaiworapuek, W. (2007). The Engineering Investigation of the Water Flow past the Butterfly Valve , Memoir - Thesis, Trinity College, Dublin, Ireland.

Del Toro, A.; Johnson M.C. and Spall, R.E. (2015). Computational Fluid Dynamics Investigation of Butterfly Valve Performance Factors.

Douglas, J.F.; Gasiorek, J.M.; Swaffield, J.A. and Jack, L.B. (2005). Fluid Mechanics, Fifth Edition, Pearson Education Limited, Harlow, England. 356.

Gite, P. (2015). Torque Optimization in Triple Offset Butterfly Valve, International Engineering Research Journal (IERJ), Special Issue 2: 5845-5848.

International Journal of Informative & Futuristic Research, 2(9): 3129 - 3139.

Kang, S.; Kim, D.; Kim, K. and Kim, J. (2014). Effect Analysis of Design Variables on the Disc ina Double-Eccentric Butterfly Valve, The Scientific World Journal, Volume, Article ID 305085, 6.

Lipták, B.G. (2006). Process Control and Optimization, Instrument Engineers’ Handbook, Fourth Edition, Taylor & Francis Group, New York, USA, 1276.

Pisal, S.K. and Sawant S.M. (2015). Testing And Performance Evaluation of Butterfly Valve, SimScale, 2018, https://www.SimScale.com/.